Computational Methods for Multi-physics Applications with Fluid-structure Interaction

Kumnit Nong[1], Eugenio Aulisa[2], Sonia Garcia[3], Edward Swim[4], and Padmanabhan Seshaiyer[*,5]
[1,5]George Mason University, [2]Texas Tech University, [3]US Naval Academy, [4]Sam Houston State University
*Corresponding author: 4400 University Drive, MS: 3F2, Science and Tech I, Department of Mathematical Sciences, George Mason University, Fairfax, VA 22030, email: pseshaiy@gmu.edu

Abstract: Efficient modeling and computation of the nonlinear interaction of fluid with a solid undergoing nonlinear deformation has remained a challenging problem in computational science and engineering. Direct numerical simulation of the non-linear equations, governing even the most simplified fluid-structure interaction model depends on the convergence of iterative solvers which in turn relies heavily on the properties of the coupled system. The purpose of this work is to model and simulate multi-physics applications that involve fluid-structure interaction using a distributed multilevel algorithm with finite elements. The proposed algorithm is tested using COMSOL which offers the flexibility and efficiency to study coupled problems involving fluid-structure interaction. Numerical results for some benchmark fluid-structure interactions are presented that validate the proposed computational methodology for solving coupled problems involving fluid-structure interaction is reliable and robust.

Keywords: Fluid-Structure Interaction; Multi-physics; Micro-Air Vehicle; Coupled Systems.

1. Introduction

The efficient solution methodology to complex multi-physics problems involving fluid-structure interactions (FSI) is a challenging problem in computational sciences. Such problems require studying complex nonlinear interactions between the structural deformation and the flow-field that often arise in applications such as blood-flow interaction with an arterial wall or computational aero-elasticity of flexible micro-air vehicles.

In the last two decades, domain decomposition techniques have become increasingly popular for obtaining fast and accurate solutions of problems involving coupled processes [1, 2, 3]. These viable domain decomposition techniques have been shown to be stable mathematically and have been successfully applied to a variety of engineering applications [4, 5, 6]. The basic idea is to replace the strong continuity condition at the interfaces between the different sub-domains by a weaker one to solve the problem in a coupled fashion. The purpose of this paper is to develop a coupled FSI algorithm and implement the algorithm using COMSOL to some benchmark FSI problems.

In section 2, we present the formulation of a one-dimensional FSI problem using an arbitrary Lagrangian Eulerian formulation and employ the finite element method via COMSOL for solving the coupled problem. In section 3, we present an optimal control formulation of the FSI problem for the 1-D problem presented in section 2. In section 4, we consider a three dimensional FSI problem with application to micro-air vehicles.

2. A Coupled One-Dimensional FSI Model Problem

For simplicity of presentation, we first develop the model for a one-dimensional FSI problem that involves a structural domain interacting with a fluid medium. The model is set up so that initially the fluid domain occupies the interval $(-1, 0)$ and the elastic structure occupies the interval $(0, 1)$. As the fluid flow deforms the adjacent solid, we allow the movement of the interface to depend on the velocity of the fluid. This is illustrated in figure 1 below.
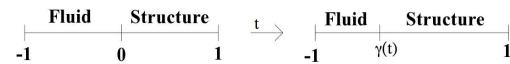

Figure 1: Undeformed and deformed computational domains

We denote by $\gamma(t)$ the position of this interface at any positive time t. For all values of x in the interval $(-1, \gamma(t))$, we model the fluid velocity v and the pressure p using a generalization of models employed by [7] using:

$$v_t - \alpha v_{xx} + (1+\beta)v v_x + \varepsilon p_x = f_f$$

1. REPORT DATE **OCT 2010**	2. REPORT TYPE	3. DATES COVERED **00-00-2010 to 00-00-2010**

4. TITLE AND SUBTITLE **Computational Methods for Multi-physics Applications with Fluid-structure Interaction**		5a. CONTRACT NUMBER
		5b. GRANT NUMBER
		5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)		5d. PROJECT NUMBER
		5e. TASK NUMBER
		5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **George Mason University,Department of Mathematical Sciences,4400 University Drive, MS: 3F2,Fairfax,VA,22030**		8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)		10. SPONSOR/MONITOR'S ACRONYM(S)
		11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES
Presented at the 2010 COMSOL Conference, 7-9 Oct, Boston, MA.

14. ABSTRACT
Efficient modeling and computation of the nonlinear interaction of fluid with a solid undergoing nonlinear deformation has remained a challenging problem in computational science and engineering. Direct numerical simulation of the non-linear equations, governing even the most simplified fluid-structure interaction model depends on the convergence of iterative solvers which in turn relies heavily on the properties of the coupled system. The purpose of this work is to model and simulate multi-physics applications that involve fluid-structure interaction using a distributed multilevel algorithm with finite elements. The proposed algorithm is tested using COMSOL which offers the flexibility and efficiency to study coupled problems involving fluid-structure interaction. Numerical results for some benchmark fluid-structure interactions are presented that validate the proposed computational methodology for solving coupled problems involving fluid-structure interaction is reliable and robust.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**Same as Report (SAR)**	**24**	

In this model, $\alpha = \alpha(\nu) > 0$ is a parameter depending on the kinematic viscosity ν of the fluid. The constants $\beta \in [0, 0.5]$ and $\varepsilon \geq 0$ vary depending on the material properties of the fluid. Here f_f is the external force on the fluid. In this work we consider the specific case of the Burger's equation using the parameter choices, $\alpha = \nu$, $\beta = 0.5$, $\varepsilon = 0$. It is well known that this choice of values yields stability of the numerical methods employed for the problem whenever no pressure term is included.

The fluid model is coupled with an elastic model that represents the structure. In particular, we consider the wave equation that models the solid displacement u of any point in the adjacent structure from its initial position given by:

$$u_{tt} - \mu u_{xx} = f_s$$

Here $\mu > 0$ and f_s is the external force on the structure. Also, the position of the interface between the two sub-domains must satisfy the movement at the interface for all times.

At the interface between the fluid and structure, we enforce continuity of the fluid velocities and the action-reaction principle:

$$v(\gamma(t), t) = u_t(0, t)$$
$$\alpha v_x(\gamma(t), t) = \mu u_x(0, t)$$

In order to account for the changing nature of the fluid, we consider the arbitrary Lagrangian-Eulerian (ALE) formulation [8]. This will allow for a dynamic computational gird that avoids extreme mesh distortion near the interface. To do this one can move the numerical grid independently of the fluid velocity on the fluid domain. Defining the grid velocity as:

$$w = \frac{x}{\gamma(t)} \dot{\gamma}(t)$$

Note that $w(\gamma(t), t) = \dot{\gamma}(t)$, $w(0, t) = 0$. Thus the grid velocity is consistent with the velocity of the fluid at the endpoints of the fluid domain. Additionally, we assume $\gamma(t) \in [-1, 1]$ for all time. The ALE form of the fluid equation that we then solve is:

$$v_t - \alpha v_{xx} + (1.5v - w)v_x = f_f$$

For the boundary conditions, we let the fluid velocity $v(-1, t) = \sin(0.1 \pi t)$ and the structural displacement $u(1, t) = 0$ for all times. For initial conditions, both the fluid and the structural domain are at rest initially.

The FSI coupled system described was implemented in COMSOL and the results of the finite element implementation of the model is summarized in figure 2 that displays the plot of the fluid velocity v and the structural velocity u_t over the time period from 0 to 1.
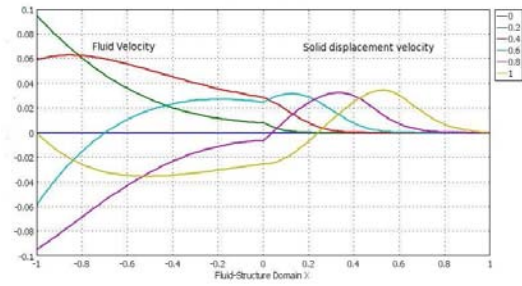

Figure 2: FSI simulation using COMSOL for time t=0 to 1.

3. Coupled FSI problem with Control

A related aspect that we consider in this work is investigating distributed control for FSI problems. In particular, one can study the coupled FSI problem using an optimal control formulation to predict the distributed control that corresponds to a prescribed velocity $\hat{v}$ and displacement data $\hat{u}$ that satisfies the boundary conditions. Specifically, we want develop a model that predicts the force that results in the minimization of the error in the fluid velocity $|v - \hat{v}|$ and the solid displacement $|u - \hat{u}|$. For simplicity, let us consider the one-dimensional model problem presented in section 2 and extend it to include control aspects.

Towards this end, let us consider the related cost functional for the associated non-linear FSI problem (note that one can similarly consider the linear FSI problem by dropping the corresponding linear terms which is not described here) given by:

$$M = \int_0^T \int_{-1}^0 \left(\frac{1}{2}(v - \hat{v})^2 + \frac{1}{2}\alpha_f f_f^2 \right)$$

$$+ \int_0^T \int_{-1}^0 \left(\frac{1}{2}(u - \hat{u})^2 + \frac{1}{2}\alpha_s f_s^2 \right)$$

$$+ \int_0^T \int_{-1}^0 \left(l(\rho_f v_t - \mu_f v_{xx} + 1.5vv_x - f_f) \right)$$

$$+ \int_0^T \int_{-1}^0 \left(g(\rho_s u_{tt} - \mu_s u_{xx} - f_s) \right)$$

where l and g are the Lagrange multipliers corresponding to the fluid velocity and the solid displacement respectively. Moreover, we also impose the continuity of the velocities over the coupled domain.

Proceeding using the standard optimal control approach of minimizing the cost function by taking the variations yields the following auxiliary system of governing equations:

In the fluid domain $-1 \le x \le 0$:

$$\rho_f v_t - \mu_f v_{xx} + 1.5vv_x - \frac{l}{\alpha_f} = 0$$

$$-\rho_f v_t - \mu_f l_{xx} - 1.5vl_x + v - \hat{v} = 0$$

In the solid domain $0 \le x \le 1$:

$$\rho_s v_t - \mu_s u_{xx} - \frac{g}{\alpha_s} = 0$$

$$\rho_s g_{tt} - \mu_s g_{xx} + u - \hat{u} = 0$$

For numerical experiments, we consider a simple solid displacement profile and the fluid velocity profile given by:

$$\hat{u} = 0.5x(x^2 - 1)t^2$$

$$\hat{v} = x(x^2 - 1)t$$

The problem was implemented in COMSOL to yield the following velocity and displacement profiles in the respective domains. This is illustrated in Figures 3 and 4 where the prescribed solution is plotted against both the solution to the both the linear and non-linear control problem.
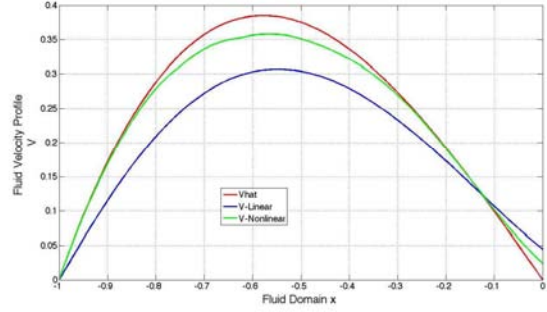

Figure 3: Displacement profile comparing the prescribed solid displacement with linear and non-linear control models.
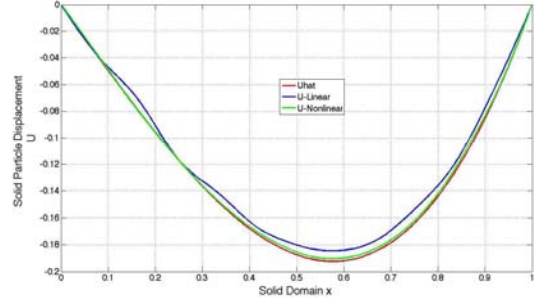

Figure 4: Fluid-velocity profile comparing the prescribed velocity with linear and non-linear control models

4. A Coupled multi-dimensional FSI problem with applications to MAVs

A Micro Air Vehicle (MAV) is a type of radio-controlled miniature aircraft that can fly at very low speeds. Due to the complexities of the wing structure of a MAV, a computational model of the aircraft wing requires a combination of many structural elements interacting with external fluid.

The wing typically consists of a flexible membrane material braced with a leading edge spar and chordwise battens (see Figure 5). The structural model must combine the model of the membrane material together with the model of the rigid battens. Most current models treat the battens as large-density membrane elements. Modeling this coupled two-dimensional structural model interacting with a three-dimensional fluid makes the problem very challenging.
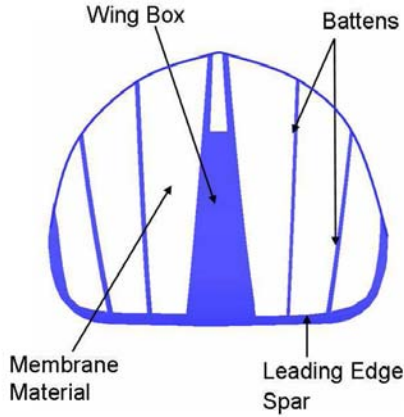
Figure 5: A simple model of a flexible MAV wing

In this work, we attempt to come up with a mathematical model that can provide insight into the dynamics of MAVs. The model presented herein is simplistic; however, one may extend this to accommodate other features.

In order to get an insight into the modeling and dynamics of a MAV, let us consider a cylindrical computational domain. In this domain we will assume that the fluid satisfies a potential equation and that a two-dimensional structural model (that will represent the MAV) is a part of one of the circular surfaces. This latter surface will represent the outflow boundary of the computational domain which is illustrated below:
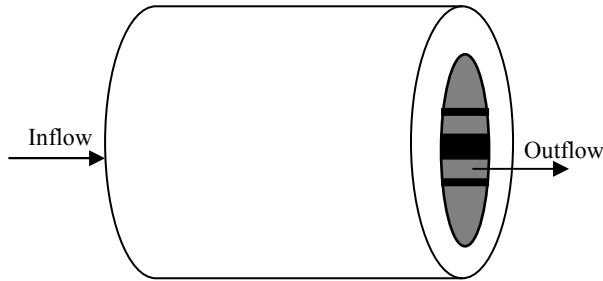
Figure 6: Computational domain for MAVs

Let the computational domain be partitioned into three sub-domains Ω_i, i=1,2,3. Let Ω_3 represent the cylinder in the computational domain where the following governing equations hold:

$$\Delta\phi = 0 \quad in \quad \Omega_3$$
$$\nabla\phi\cdot\vec{n} = 0 \quad on \quad \Gamma_f^N$$

Here Γ_f^N corresponds to the lateral surface of the cylinder where Neumann boundary conditions are prescribed.

The outflow part of the computational domain consist of the following sub-domains: Γ_f^O corresponds to the outflow region that is not a part of the structural domain; Ω_2 corresponds to the structural domain that involves the three battens; Ω_1 corresponds to the structure (shaded grey) that does not involve the three battens. We will assume the following adsorption condition:

$$\nabla\phi\cdot\vec{n} = -a\,\phi_t \quad on \quad \Gamma_f^O$$

where a is a constant. Also, Γ_f^I corresponds to the inflow surface where we prescribe:

$$\nabla\phi\cdot\vec{n} = -0.1 + 0.025\sin(2\pi t) \quad on \quad \Gamma_f^I$$

For the structural model of the MAV that is modeled via the sub-domains Ω_1 and Ω_2 we consider the following governing membrane equations for the deflection of the membrane w:

$$\rho_0\,w_{tt} - E_0\,\Delta w = f \qquad in \quad \Omega_1$$
$$(\rho_0 + \rho_1)w_{tt} - E_0\,\Delta w + E_1\,v_{yy} = -\rho_f\,\phi_t \quad in \quad \Omega_2$$
$$v = w_{yy} + \varepsilon\,\Delta v \qquad in \quad \Omega_1\cup\Omega_2$$

where E_0, E_1 are the constants corresponding to the elastic modulus and second moment of area for each of the sub-domains Ω_1 and Ω_2 respectively. Also, ρ_0, ρ_1 are the respective densities of the membrane and the battens. It must also be pointed out that half of the MAV edge was kept rigid to reflect the leading edge spar. The two systems are also coupled through the continuity of the velocities:

$$\nabla\phi\cdot\vec{n} = w_t \quad on \quad \Omega_1$$

The fully coupled system described herein was modeled and solved in COMSOL and the results for the membrane deflection are shown in Figures 7 and 8.
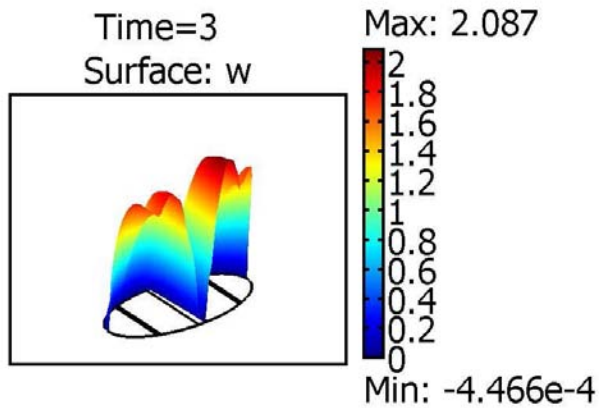
Figure 7: Three-dimensional MAV
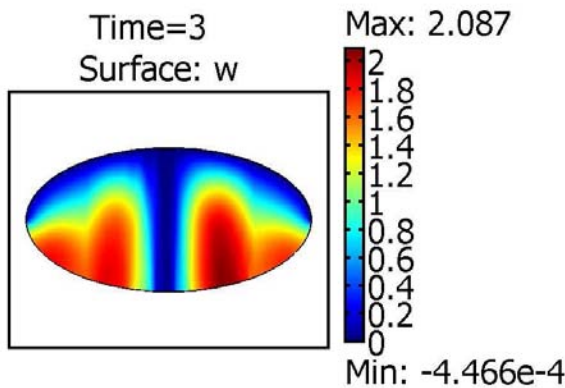membrane wing deflection

Figure 8: Two-dimensional MAV
membrane wing deflection

5. Conclusions

In this work, a coupled computational methodology to solve problems that involve fluid-structure interaction has been presented for various benchmark problems. The problems considered in this work included a one dimension problem coupling fluid and structure with and without control and an application problem in three dimensions involving MAVs. The one dimensional problems provide a great insight into the nature of the coupled behavior of the interaction between the fluid velocity and the structural displacement. The importance of the non-linear term in the fluid equations was illustrated in the control problem that helped decrease the error between the prescribed and computed solutions.

6. References

1. Seshaiyer, P. and Suri, M., hp submeshing via non-conforming finite element methods, *Computer Methods in Applied Mechanics and Engineering*, 189, 1011–1030, (2000).
2. Ben Belgacem, F., Chilton, L.K. and Seshaiyer, P., The hp-Mortar Finite Element Method for Mixed elasticity and Stokes Problems, *Computers and Mathematics with Applications,* 46, 35–55, (2003).
3. Swim, E.W. and Seshaiyer, P., A nonconforming finite element method for fluid-structure interaction problems, *Computer Methods in Applied Mechanics and Engineering*, 195, 2088–2099, (2006).
4. Aulisa, E., Manservisi, S. and Seshaiyer, P., A computational multilevel approach for solving 2D Navier-Stokes equations over non-matching grids, *Computer Methods in Applied Mechanics and Engineering*, 195, 4604–4616, (2006).
5. Aulisa, E., Manservisi, S. & Seshaiyer, P., A non-conforming computational methodology for modeling coupled problems, *Nonlinear Analysis*, 6, 1445–1454 (2005).
6. Aulisa, E., Manservisi, S. and Seshaiyer, P., A multilevel domain decomposition approach to solving coupled applications in computational fluid dynamics, *International Journal for Numerical Methods in Fluids*, 56, 1139–1145 (2008).
7. C. Grandmont, V. Guimet and Y. Maday. Numerical analysis of some decoupling techniques for the approximation of the unsteady fluid structure interaction. *Mathematical Models and Methods in Applied Sciences*, 11: 1349-1377, (2001).
8. J. Donea, S. Giuliani and J. Halleux. An arbitrary Lagrangian-Eulerian finite element method for transient fluid-structure interactions. *Computer Methods in Applied Mechanics and Engineering*, 33:689-723, (1982).

7. Acknowledgements

Computational Methods for Multi-physics Applications with Fluid-structure Interaction

Kumnit Nong and Padmanabhan Seshaiyer
Department of Mathematical Sciences
George Mason University

Eugenio Aulisa
Department of Mathematics and Statistics
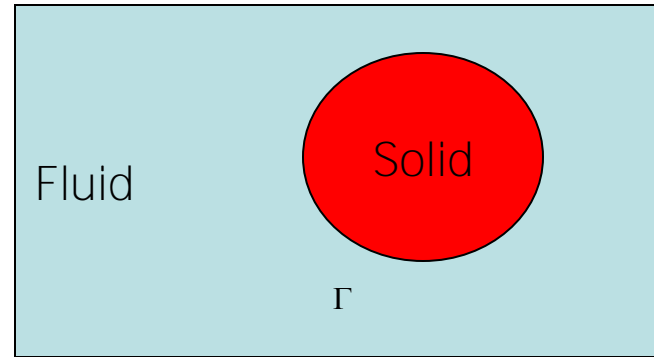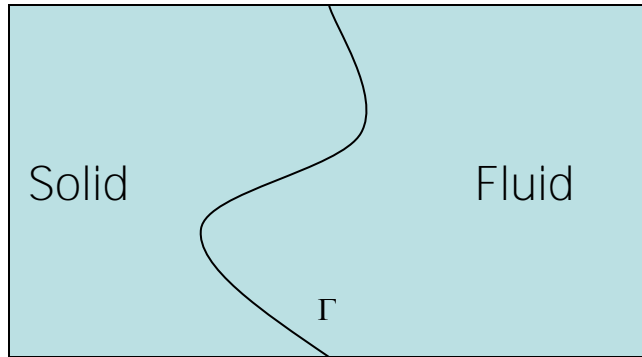Texas Tech University

Sonia Garcia
Department of Mathematics
United States Naval Academy

Edward Swim
Department of Mathematics
Sam Houston State University

COMSOL 2010
Boston, USA
(October 8, 2010)

Flow-structure interaction (FSI)

$$Fluid : \Omega_f \times (0,T)$$

$$\rho_f \frac{\partial \vec{u}}{\partial t} - \nu \Delta \vec{u} + (\vec{u}.\nabla)\vec{u} + \nabla p = \vec{f}$$

$$\nabla . \vec{u} = 0$$

$$Solid : \Omega_s \times (0,T)$$

$$\rho_s \frac{\partial^2 w}{\partial t^2} - \nabla . \widetilde{\sigma} = \vec{b}$$

$$\widetilde{\sigma} = \lambda \, tr(\widetilde{\varepsilon}) + 2\mu\,\widetilde{\varepsilon}$$

$$\widetilde{\varepsilon} = 0.5[\nabla w + (\nabla w)^T]$$

Fluid-Structure Interaction (1D)

Fluid   Structure   Fluid   Structure

0   1   2   0 $\gamma(t)$ 2
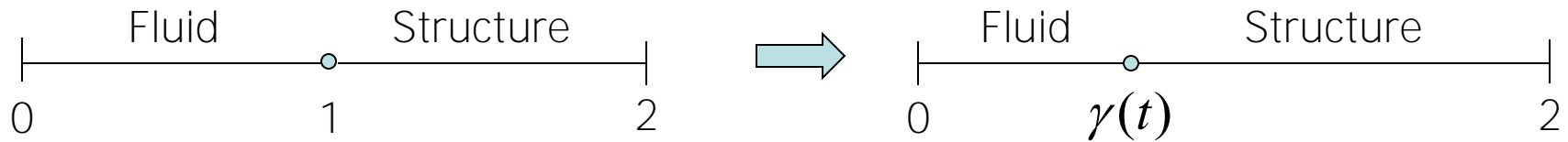
$$\rho_f \frac{\partial u}{\partial t} - \mu_f \frac{\partial^2 u}{\partial x^2} + \frac{3}{2} \rho_f u \frac{\partial u}{\partial x} = f(t,x) \qquad x \in (0,1)$$

$$\rho_s \frac{\partial^2 d}{\partial t^2} - \mu_s \frac{\partial^2 d}{\partial x^2} \qquad\qquad = g(t,x) \qquad x \in (1,2)$$

$$\gamma(t) = 1 + d(t,1) \qquad t \geq 0$$

$$u(t,\gamma(t)) = \frac{\partial d}{\partial t}(t,1)$$

$$\mu_f \frac{\partial u}{\partial x}(t,\gamma(t)) = \mu_s \frac{\partial d}{\partial x}(t,1)$$

ALE Formulation

$$w(t,x) = \frac{x}{\gamma(t)} \, \dot{\gamma}(t) \qquad x \in (0, \gamma(t))$$

For all $\xi \in (0, \gamma(s))$:

$$\frac{dx_s}{dt}(t,\xi) = w(t, x_s(t,\xi))$$

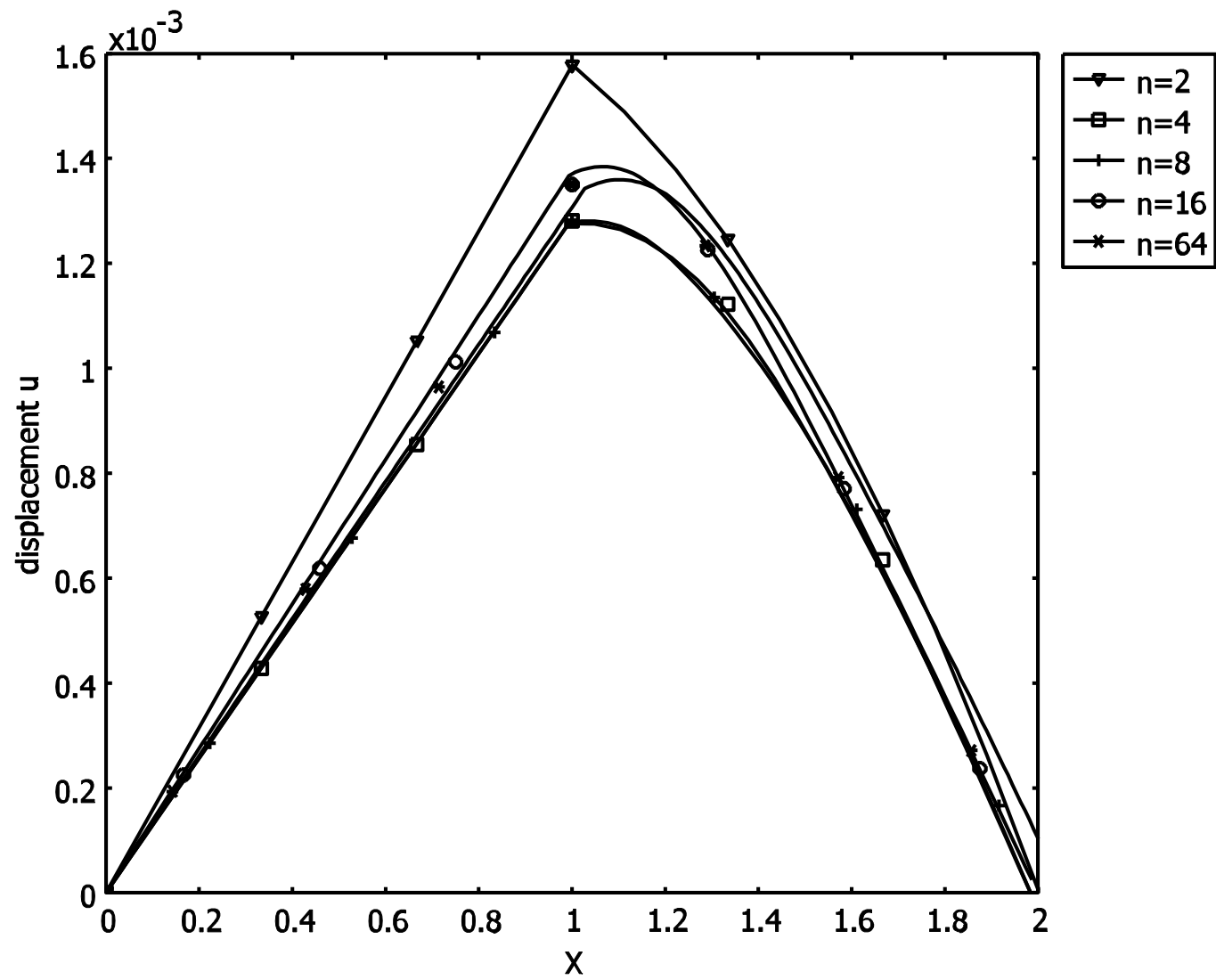$$x_s(s,\xi) = \xi$$

$$x_s(t,\xi) = \xi \frac{\gamma(t)}{\gamma(s)}$$
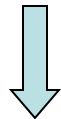
$$\frac{\partial v}{\partial t}(t,\xi) - \mu_f \frac{\partial^2 u}{\partial x^2}(t, x_s(t,\xi)) + [1.5u - w]\frac{\partial u}{\partial x}(t, x_s(t,\xi)) = f(t, x_s(t,\xi))$$
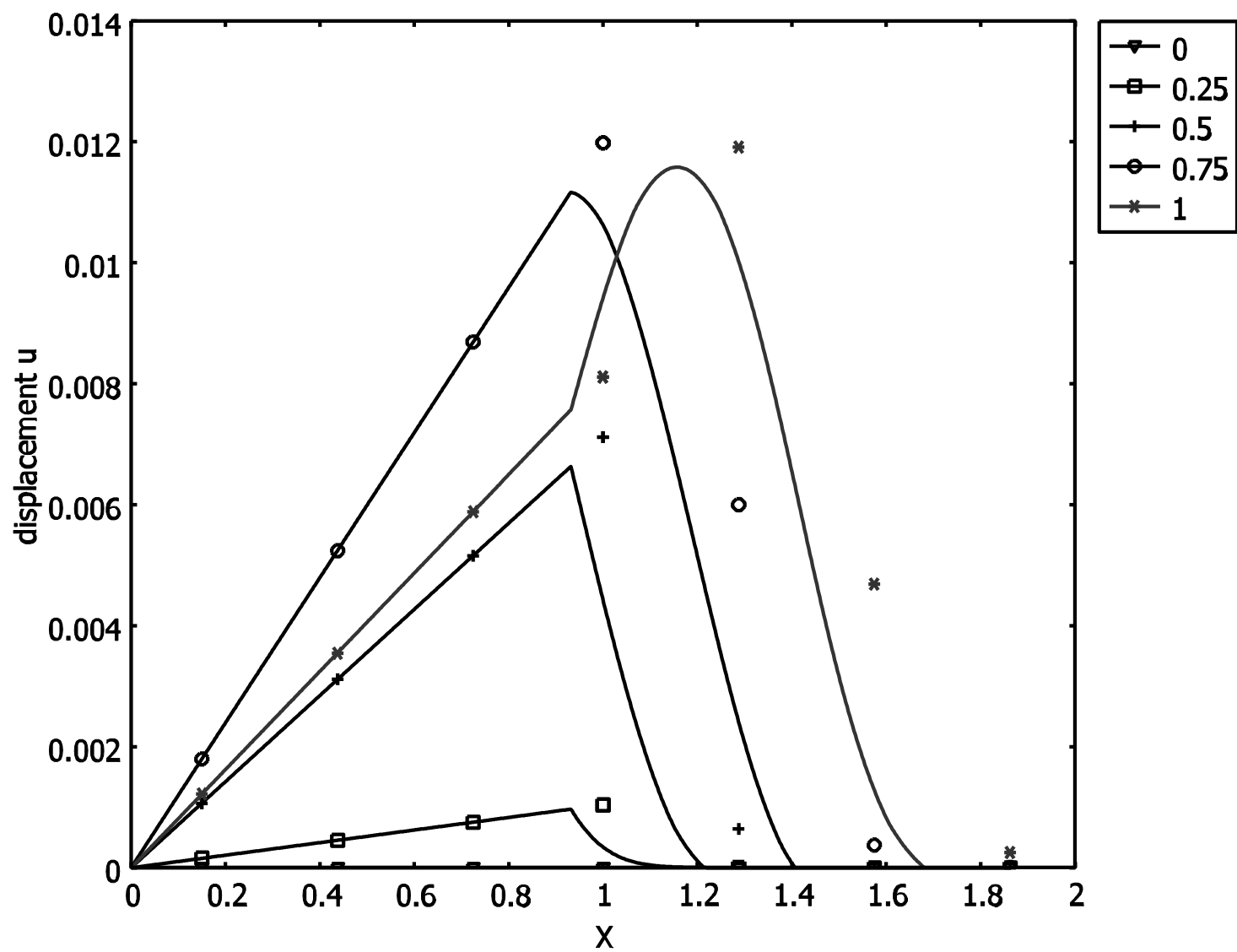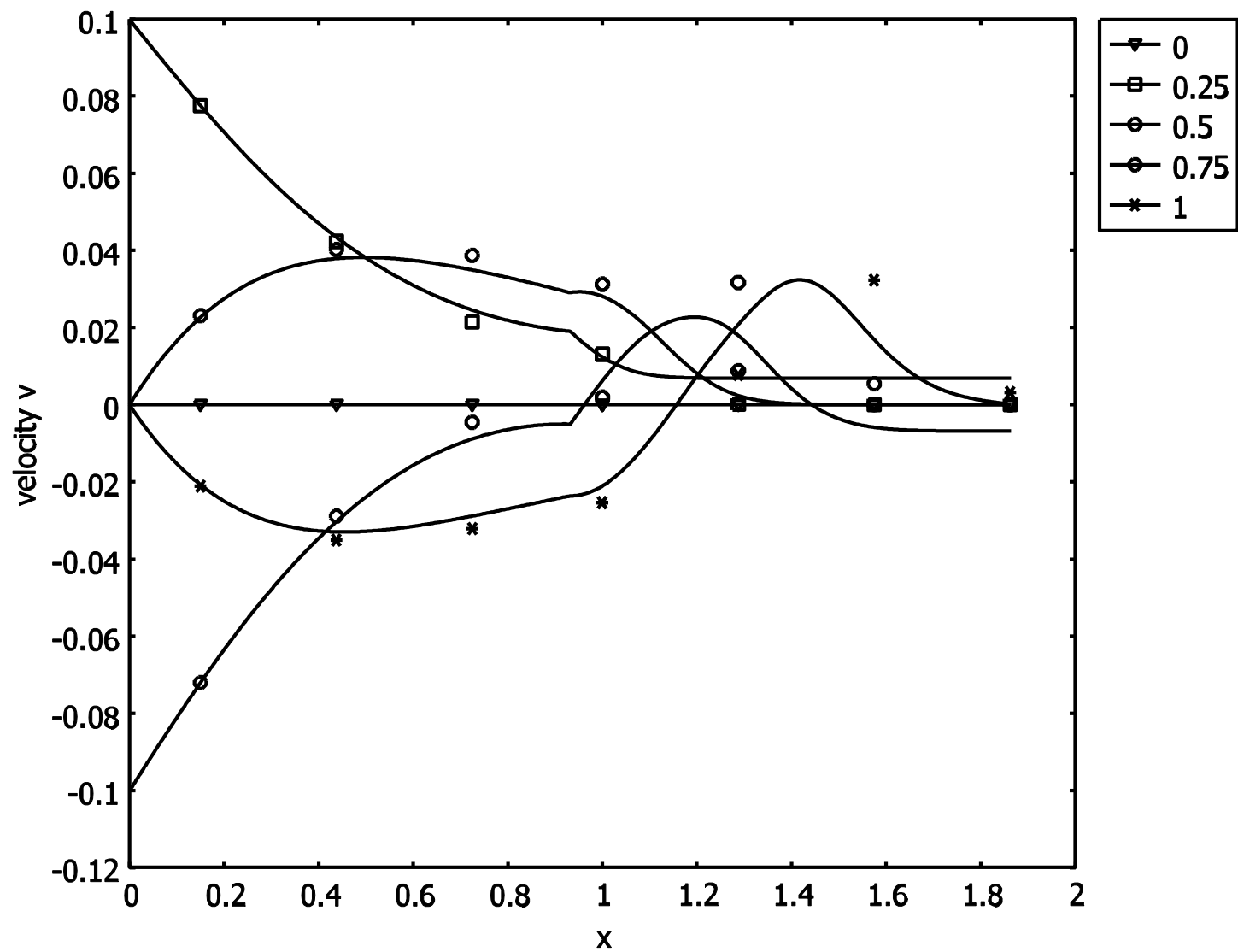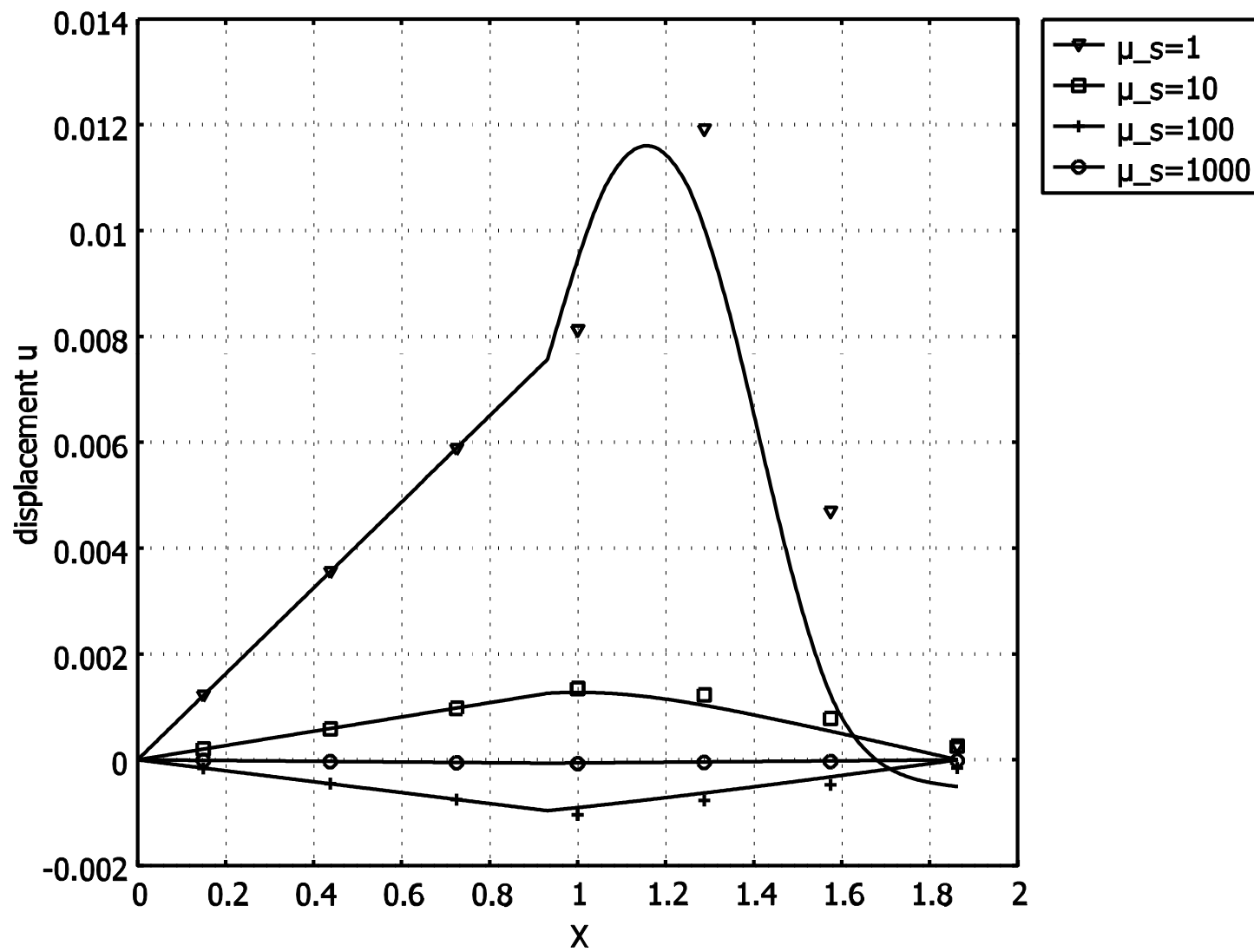
Coupled FSI with Control

$$M = \int\limits_0^T \int\limits_{-1}^{\gamma(t)} \left(\frac{1}{2}(u - \hat{u})^2 + \frac{1}{2}\alpha_f\, f_f^2 \right)$$

$$+ \int\limits_0^T \int\limits_{-1}^0 \left(\frac{1}{2}(d - \hat{d})^2 + \frac{1}{2}\alpha_s\, f_s^2 \right)$$

$$+ \int\limits_0^T \int\limits_{-1}^{\gamma(t)} \left(l(\rho_f u_t - \mu_f u_{xx} + 1.5 u u_x - f_f) \right)$$

$$+ \int\limits_0^T \int\limits_{-1}^0 \left(g(\rho_s d_{tt} - \mu_s d_{xx} - f_s) \right)$$

Auxiliary system of PDEs

- In the fluid domain:

$$\rho_f u_t - \mu_f u_{xx} + 1.5 u u_x - \frac{l}{\alpha_f} = 0$$

$$-\rho_f u_t - \mu_f l_{xx} - 1.5 u l_x + u - \hat{u} = 0$$

- In the solid domain:

$$\rho_s u_t - \mu_s d_{xx} - \frac{g}{\alpha_s} = 0$$

$$\rho_s g_{tt} - \mu_s g_{xx} + d - \hat{d} = 0$$

Fluid-velocity Profile

$$\hat{d} = 0.5x(x^2 - 1)t^2$$
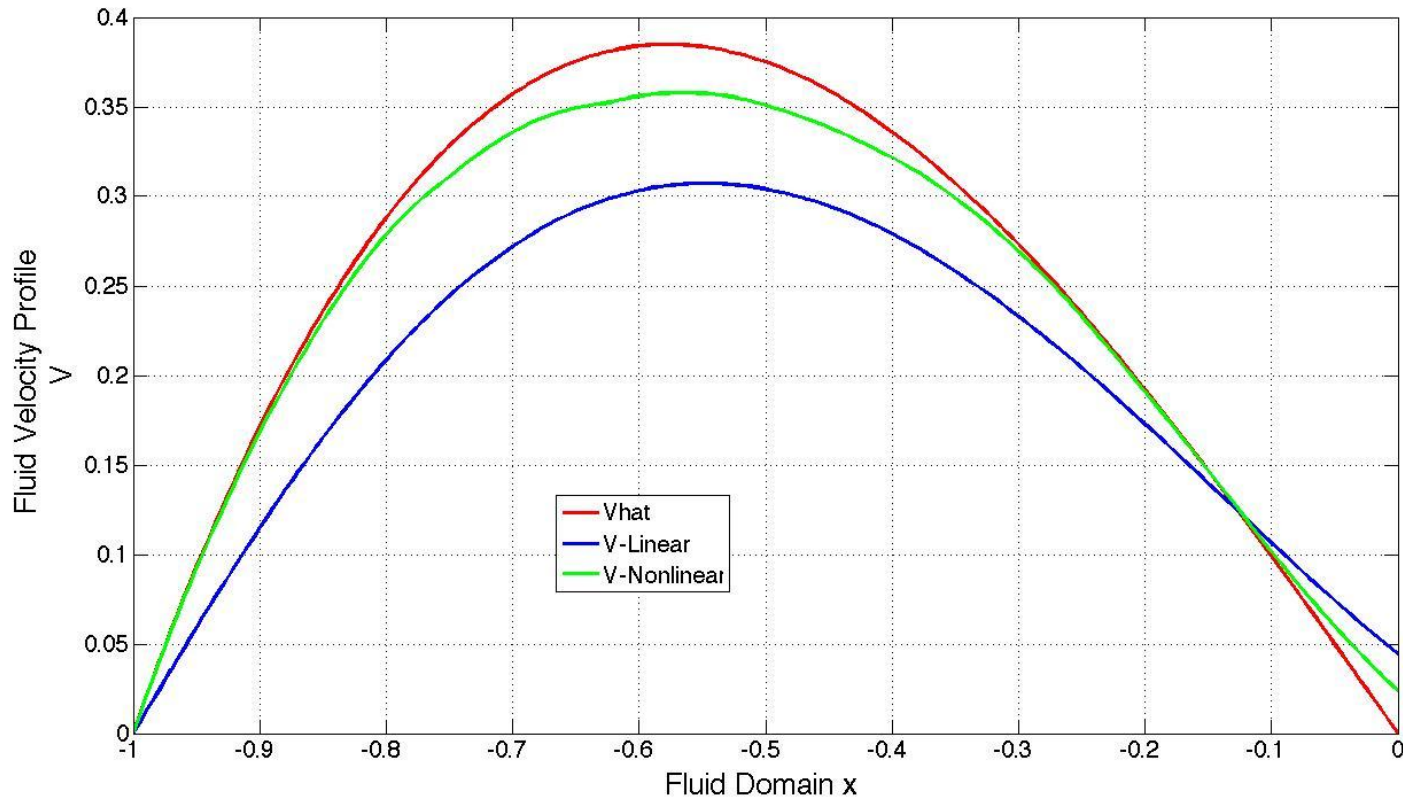
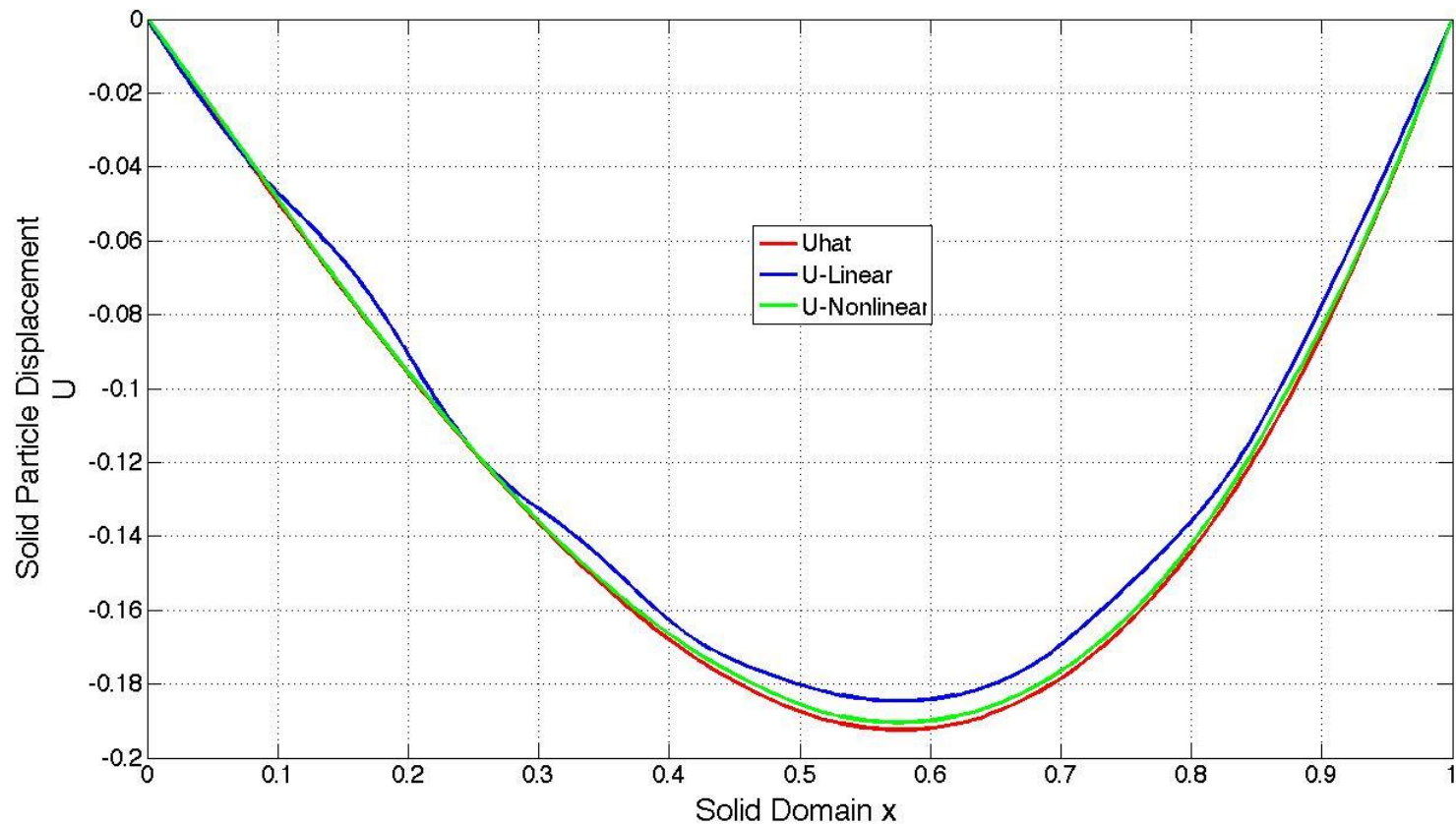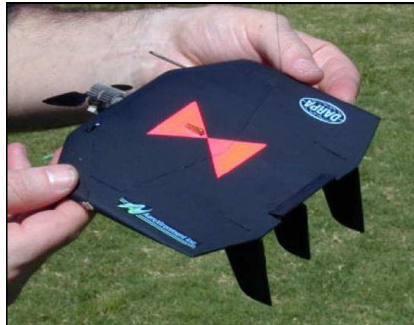$$\hat{u} = x(x^2 - 1)t$$

Displacement Profile

$$\hat{d} = 0.5x(x^2 - 1)t^2$$

$$\hat{u} = x(x^2 - 1)t$$

MAV: Membrane Wing Deflection

Computational modeling of highly flexible membrane wings for MAVs
Ferguson, L., Aulisa, E., Seshaiyer P. and Gordnier R., (AIAA 2006-1661)
Computational modeling of coupled membrane-beam flexible wings for MAVs
L. Ferguson, P. Seshaiyer, R. Gordnier, P. Attar (AIAA 2007-1787)
Nonlinear Models for Biologically- Inspired Elastic Membrane Wings
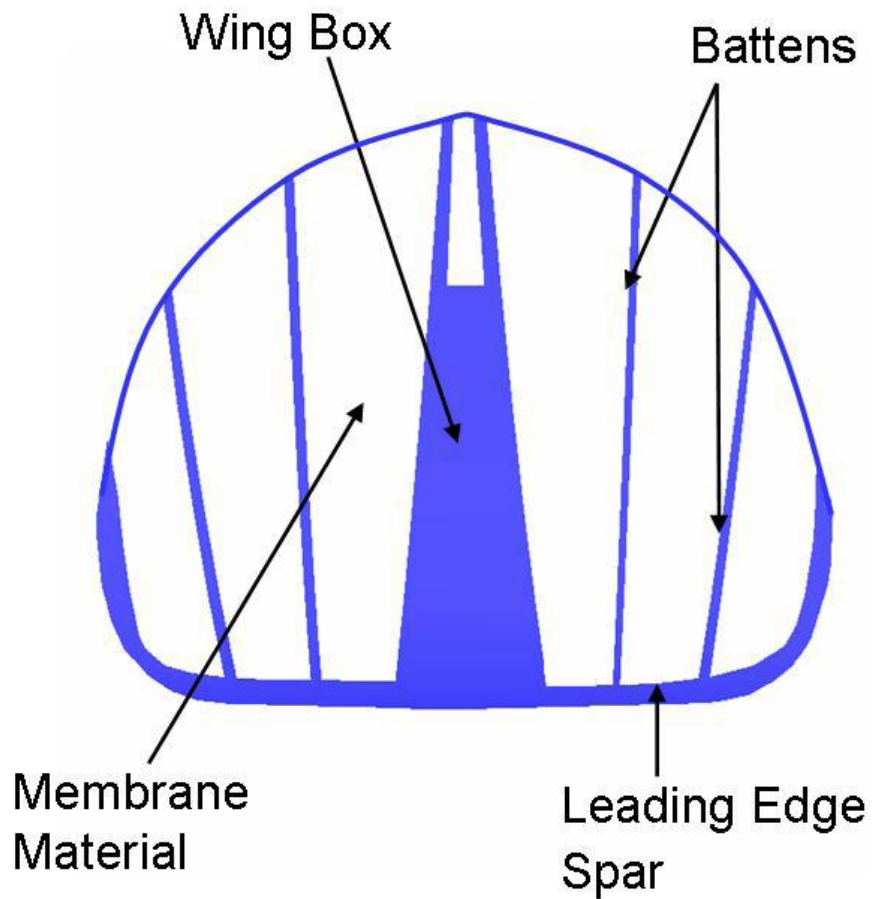E. Swim and P. Seshaiyer (AIAA-2008-2008)

Experimental Challenges in MAV Design

- Small size
- High surface-to-volume ratio
- Constrained weight and volume limitations
- Low Reynolds number regime
- Low aspect ratio fixed to rotary to flapping wings
- Longer flight time
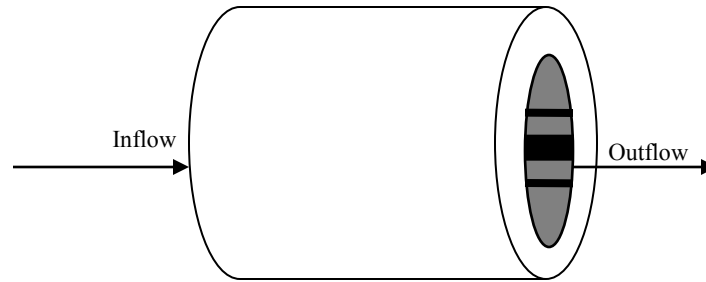- Better range-payload performance

Computational Challenges in MAV Design
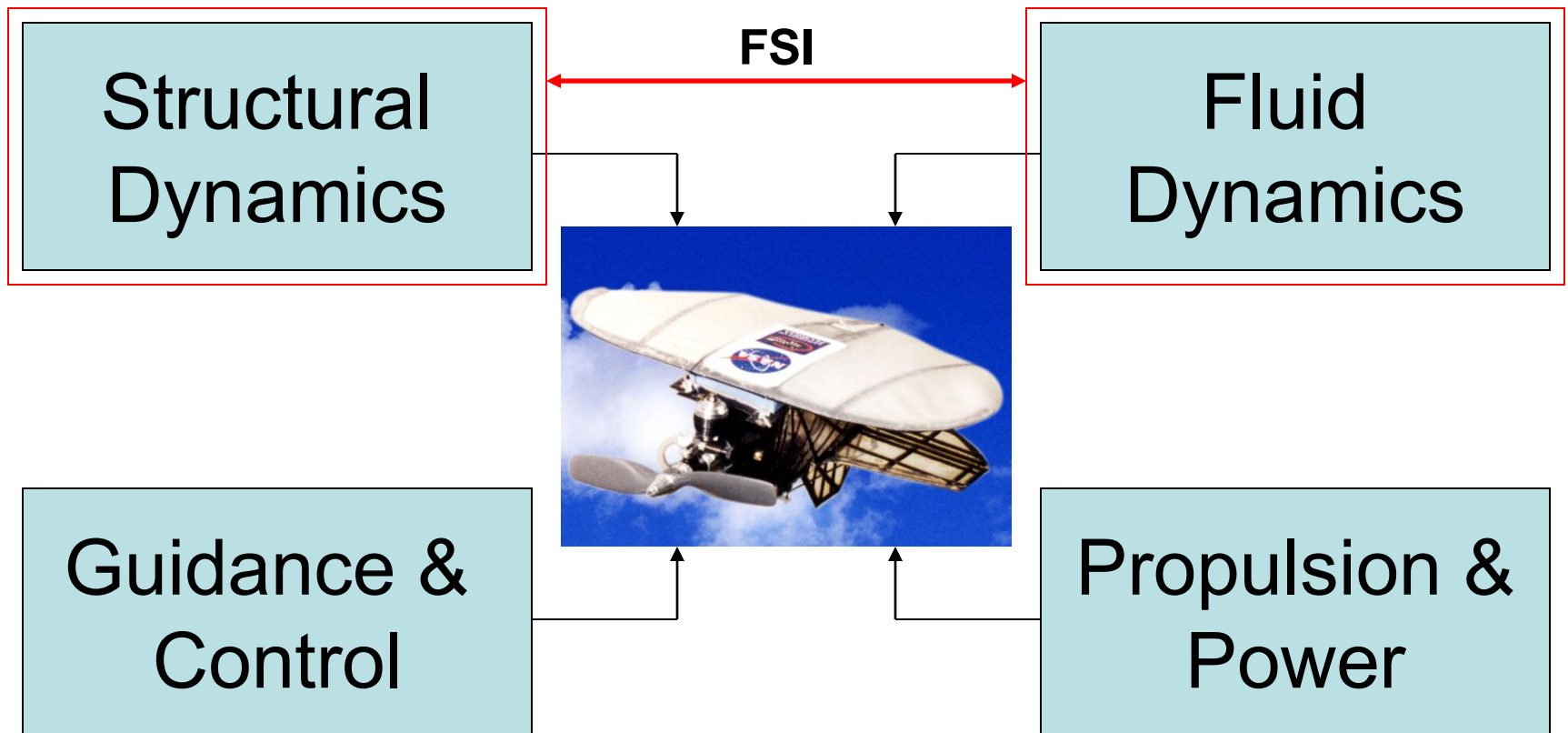
Model of a flexible MAV wing

Computational Model for MAV

$$\Delta\phi = 0 \quad in \quad \Omega_3$$
$$\nabla\phi \cdot \vec{n} = 0 \quad on \quad \Gamma_f^N$$

$$\rho_0 \, w_{tt} - E_0 \, \Delta w = f \qquad in \quad \Omega_1$$
$$(\rho_0 + \rho_1) w_{tt} - E_0 \, \Delta w + E_1 \, v_{yy} = -\rho_f \, \phi_t \quad in \quad \Omega_2$$
$$v = w_{yy} + \varepsilon \, \Delta v \qquad in \quad \Omega_1 \cup \Omega_2$$

$$\nabla\phi \cdot \vec{n} = -a \, \phi_t \quad on \quad \Gamma_f^O$$

$$\nabla\phi \cdot \vec{n} = -0.1 + 0.025\sin(2\pi t) \quad on \quad \Gamma_f^I$$

$$\nabla\phi \cdot \vec{n} = w_t \quad on \quad \Omega_1$$

Membrane Wing Deflection

Membrane Wing Deflection